Vertex-Based Finite-Volume Solution of the Two-Dimensional Navier-Stokes Equations

Sunil Kumar Chakrabartty*
National Aeronautical Laboratory, Bangalore, India

Introduction

PAST experiences in solving inviscid flows by the Euler equations provide many opportunities for exploring possible Navier-Stokes solvers. The finite-volume spatial discretization with the Runge-Kutta time-stepping scheme developed for the Euler equations has been successfully extended by Swanson and Turkel[1] to the computation of viscous flows. Their formulation for the finite-volume scheme is of the cell-centered type, where the flow quantities are associated with the center of a cell in the computational mesh and the fluxes across the cell boundaries are calculated using arithmetic means of values in the adjacent cells. The main advantage of the finite-volume method is its flexibility of treating arbitrary geometries. The cell-centered scheme loses its accuracy with grid stretching and skewness.[2] The nodal point discretization, where the flow quantities are ascribed to the corners of the cell, can give better accuracy for the highly stretched and skewed grids[2,3] that are necessary for viscous flow computations. The surface boundary conditions can be satisfied exactly at the vertices along the body surface, and the pressure on the wall can be computed directly by this scheme, whereas an extrapolation is necessary if one uses the cell-centered scheme. A nodal point finite-volume space discretization scheme[3,4] has been used here to solve the two-dimensional Reynolds-averaged Navier-Stokes equations with a thin-layer type of approximation and a simple two-layer algebraic[5] eddy viscosity model. In the present work, the efficiency of the Runge-Kutta scheme and the benefit of convergence acceleration techniques have also been utilized. The results obtained for turbulent flow past a NACA 0012 airfoil have been compared with available numerical and experimental results.

Governing Equations and Boundary Conditions

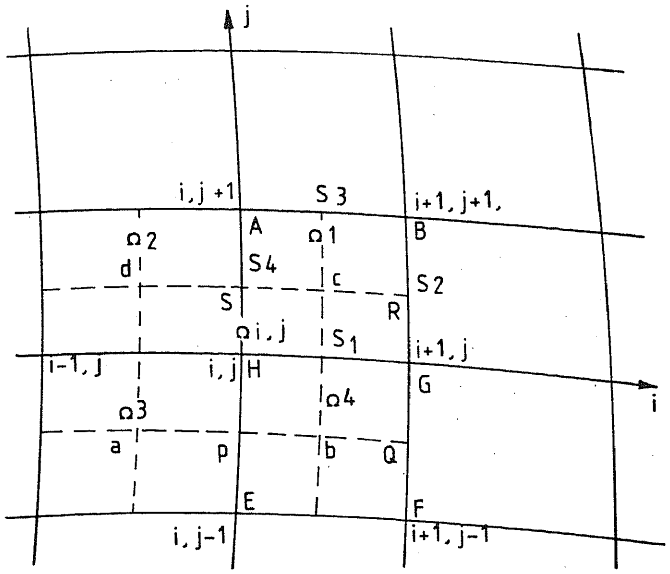
The Navier-Stokes equations representing the conservation laws have been considered here in integral form[1] for its ability to treat flow discontinuities automatically. To complete the set of equations for a compressible fluid, a thermodynamic equation of state, Stokes' hypothesis, and Sutherland's law have been considered along with the constant Prandtl number assumption.[3,4]

For turbulent flows the compressible Reynolds-averaged Navier-Stokes equations exhibit a term by term correspondence with their laminar flow counterparts, except that the stress tensor is augmented by the Reynolds stress tensor and the heat flux vector is augmented by the additional turbulence heat flux. To close the time-averaged equations in turbulent flow, the two-layer algebraic eddy viscosity model of Baldwin and Lomax[5] has been used.

The boundary conditions are that the velocity components must be zero (no slip) at the body surface, and the wall temperature is either prescribed or its normal derivative is zero (adiabatic wall, present case). A continuity condition has been

Received Jan. 28, 1989; revision received Dec. 5, 1989.

*Scientist, Computational and Theoretical Fluid Dynamics Division.



Figs. 1 Finite-volume mesh for nodal point scheme.

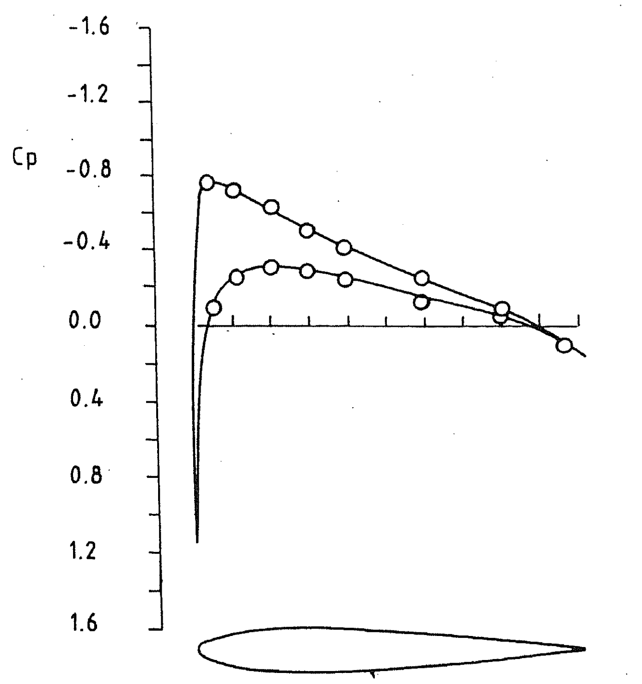


Fig. 2 Comparison of pressure distribution for turbulent flow past a NACA 0012 airfoil; $M_\infty = 0.50$, $\alpha = 1.77$ deg, $Re = 2.91 \times 10^6$

Table 1 Comparison of aerodynamic coefficients for turbulent flow over a NACA 0012 airfoil[a]

Method	Remarks	Mesh	CL	CD	CM
Ref. 1 C-type	Finite-volume Runge-Kutta	120×50	0.198	—	—
Ref. 6	Experiment		0.195	0.0070	—
Present C-type	Without effect of vortex at far field; 1500 iterations	131×61	0.1929	0.0131	0.0070
	With effect of vortex at far field, 2000 iterations	131×61	0.2071	0.0128	0.0072
	Without effect of vortex at far field, 1000 iterations	165×61	0.1978	0.0096	0.0064
	Without effect of vortex at far field, 4000 iterations	165×61	0.208	0.0081	0.0058

[a] $M_\infty = 0.50$, $\alpha = 1.77$ deg, $Re = 2.91 \times 10^6$

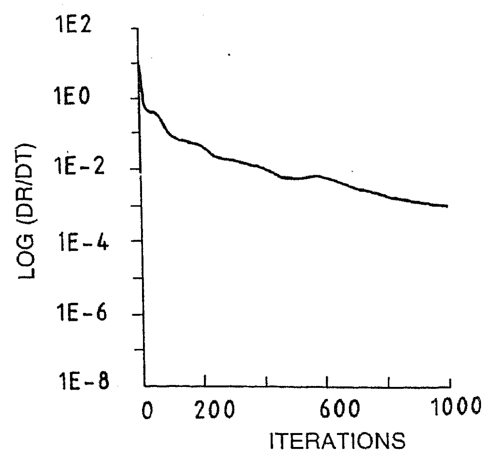


Fig. 3 Convergence history for turbulent flow past a NACA 0012 airfoil; $M_\infty = 0.50$, $\alpha = 1.77$ deg, $Re = 2.91 \times 10^6$ (165×61 grid).

used along the cut boundary in the computational domain. At the far-field boundary, the viscous effects are assumed to be negligible. The treatment of the far-field boundary in the present analysis is based on Riemann invariants for the one-dimensional flow normal to the boundary. To reduce the extent of the far-field boundary in an asymmetric flow with circulation, the following modification has also been tried here. The effect of a single vortex in compressible medium centered at the airfoil has been added to the freestream flow, and the modified freestream values have been added to compute the Riemann invariants.[4]

Solution Process and Algorithm

To solve the Navier-Stokes equations numerically, a semi-discretization is used that completely separates the discretization of space and time derivatives. To simulate the wake in a better way, the present computations are carried out with algebraically generated C-type body-fitted grids. Once the Cartesian coordinates of the four vertices of every cell are given, the Euler fluxes across the four neighboring cells Ω_1, Ω_2, Ω_3, and Ω_4 can be calculated directly from the flow variables defined at corner points (Fig. 1). To compute the viscous fluxes, Green's theorem has been used to get the first derivatives at the midpoints of the cell boundaries. The scheme can be summarized as follows:

1) Calculate first derivatives of all of the flow variables at $(i + \frac{1}{2}, j)$ using the full control volume $PQRS$ for all (i,j).

2) Calculate the stress tensors and flux terms.

3) Find the difference of flux quantities across two surfaces AB and HG to get the viscous flux over the cell $HGBA$.

4) Add these to the corresponding Euler fluxes for the same cell.

5) Take the average of the four neighboring cells of the point (i,j) to get the net flux across $\Omega_{i,j}$.

It is to be noted that streamwise-like differences are neglected at the second stage in this process (neglecting the diffusion), where in Ref. 1 these were neglected even at the first stage (thin-layer approximation). The resulting system of ordinary differential equations in time is then solved using an explicit five-stage Runge-Kutta time-stepping scheme. Second- and fourth-order artificial dissipation terms[1] have been added for numerical stability. To accelerate the rate of convergence, local time stepping, enthalpy damping, and residual smoothing[1,3] have been applied.

Results and Discussion

Turbulent flow past a NACA 0012 airfoil has been considered at freestream Mach number $M_\infty = 0.5$, angle of attack $\alpha = 1.77$ deg, and freestream Reynolds number $Re = 2.91 \times 10^6$. In the present computation, 165×61 cells were taken with minimum height of the cell near the boundary $Y_{min} = 0.0005$, such that 10–15 cells were accommodated inside the boundary layer. The CPU time taken for the 1000 iterations required for the solution to converge on a CRAY-1 computer is 428 s. The effect of using a vortex at the airfoil for the far-field boundary condition is shown in Table 1 along with a comparison of the aerodynamic coefficients. In all of the computations made so far, no noticeable difference has been observed in the final pressure distribution on the airfoil. Figure 2 shows a representative pressure distribution with good agreement between the computed and experimental results.[6] The incorporation of the vortex results in a higher lift. Oscillatory behavior was observed in convergence histories for 131×61 mesh points in both the cases of with and without considering the effect of a vortex in the far field after about 500 iterations. A smooth convergence has been achieved by using 165×61 mesh points without considering the effect of the vortex. The computations were continued up to 4000 iterations, and the convergence history is shown in Fig. 3. Consideration of the effect of a vortex leads to an unsteady solution in this case.

Acknowledgment

This work contains a part of the work done by the author from November 1986 to August 1987 at the Institute of Design Aerodynamics, DFVLR, Braunschweig, under a joint collaboration program between the Council of Scientific and Industrial Research (India) and Deutsche Forschungs und Versuchsanstalt für Luft und Raumfahrt (West Germany). The valuable discussions the author had with Dr. Anand Kumar and Dr. P. K. Dutta of the National Aeronautical Laboratory, Bangalore, are gratefully acknowledged.

References

[1] Swanson, R. C., and Turkel, E., "A Multistage Time Stepping Scheme for the Navier-Stokes Equations," AIAA Paper 85-0035; Jan. 1985.

[2] Rossow, C., "Comparison of Cell Centre and Cell Vertex Finite Volume Schemes," *Proceedings of 7th GAMM Conference,* 1987.

[3] Chakrabartty, S. K., "Numerical Solution of Two-Dimensional Navier-Stokes Equations by Finite Volume Method," National Aeronautical Lab., Bangalore, India, NAL-PD-FM 8726, also DFVLR-IB-129-87/23, 1987.

[4] Chakrabartty, S. K., "Numerical Solution of Navier-Stokes Equations for Two Dimensional Viscous Compressible Flows," *AIAA Journal,* Vol. 27, No. 7, 1989, pp. 843–844.

[5] Baldwin, B. S., and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-257, Jan 1978

[6] Thibert, J. J., Granjacques, M., and Ohman, L. H., "NACA 0012 airfoil," Experimental data base for computer program assessment, AGARD-AR-138, May 1979.